International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 07 | July 2021 www.irjet.net p-ISSN: 2395-0072

CAE ANALYSIS OF SIDE IMPACT ON FRONT DOOR TRIMS

Suyog M Dandge[1]

[1]BE, Dept. of Mechanical Engineering, MVP's KBT College of Engineering, Nashik, Maharashtra (India).

---***---

Abstract – *Safety is a major concern for automobile users as high-speed cars are introduced in markets. In today's scenario, 90% of cost paid by customer is for their life support and safety systems. India losses 3% of its GDP to road accidents. Car body, door steel and door panel play crucial role in protecting occupants from external impacts. Strength, elasticity, energy absorbing capacity and deformation rate of material affects severity of accidents. Due to which Crashworthiness got popular in recent decades. Side impact is found to be more severe and effective in comparison with front, top and rear impact. Automotive industries are also focusing on optimizing weight of door structures in order to increase fuel efficiency of vehicle without compromising its shock absorbing capacity. Every industry has to pass standard impact test prescribed by authorized institution in order to introduce their vehicle in market. In early evolution of automotive industry direct physical tests were conducted on trial and error method basis, which consumes much cost, time and auxiliary resources. In recent advancements in computerized simulation has eliminated all drawbacks and provided cost effective solutions which got popular in recent decades. Software use has enabled designers to predict results of impact and modify their design accordingly. Our problem statement focuses on side impact simulation using software tools like Hypermesh, LS-Dyna, etc. Mesh quality of every component plays crucial part in results. Software results are to be compared with previous results of research conducted and industrial results. Use of software tools will expose us to industrial design approach and methodology. This study constitutes meshing and analysis.*

Key Words*: *external impacts, crashworthiness, automotive industry, computerized simulation, meshing, analysis.

1. INTRODUCTION

Side collisions are common in numerous traffic incidents, and they frequently result in extremely dangerous crashes. Because there is a much smaller crash zone for absorbing energy in the side of the cars than there is in the front, side impacts require more care. The front and back structures, as well as the occupants, are almost entirely within the collision zone, resulting in serious injuries. The region around the door of the vehicle is subject to large deflections into the driving personnel's space during side impact, it is therefore necessary to produce a standard „worst case scenario‟ design criteria that a design can be produced for and attempt to reduce the risk of injury to the occupants. The different regions of injuries are such as abdomen zone, thorax region and pelvis region. In period of development stage of the car, it is very important problems to the vehicle industry that the side impact crashworthiness is forecasted. Now days the normal practice is CAE modeling and analyzing the methods. Using CAE based design modifications can be evaluated quickly to reduce the overall development time and cost of door panel assembly for side improvements to achieve target impact response. Using this CAE approach based design modifications can be evaluated quickly to reduce the overall development time and cost of door panel assembly for side impact requirements avoiding surprise failures in tests. In order to obtain optimize solution we are using the LS-Dyna which is an analysis module comes under the CAE. LS-DYNA is an advanced general-purpose multiphysics simulation software package. LS-DYNA supports analyzing and revealing unknown phenomena to achieve an optimal manufacturing technique, in both research and actual manufacturing. LS-DYNA can resolve accurately many difficult phenomena seen during manufacturing. "Nonlinear" problems, including large deformations, nonlinear material and contact can be solved robustly, accurately, and quickly. Computer simulation of crash events has become a valuable and necessary tool in the design and improvement of automotive structures for better crashworthiness behavior. Crashworthiness analysis has evolved through rapid improvements in computer technology, as well as through theoretical improvements in solid mechanics concerning large deformation dynamics and contact. Significant applications involve full car analysis such as frontal, rear, and side impact. During the past decade, reducing the severity of side-impact collisions has been an emerging area of research by a variety of organizations and research communities. The side impact test and evaluation criteria to reduce the severity of vehicle-to-vehicle side-impact collisions. Similarly, the international research community has developed test procedures for performing impacts into poles, one of the most severe types of side-impact collisions. Preliminary side-impact test and evaluation procedures have been conducted for roadside safety hardware, like guardrails, guardrail terminals, luminaries‟ supports, utility poles, and signs. Recommendations for performing roadside hardware side-impact crash tests are summarized; the results of several side-impact roadside hardware crash tests are described; the proposed test and evaluation procedures are compared with other major side-impact test and evaluation procedures.

2. ANALYSIS

2.1 Analysis Approach:

As the main objective of the project is related with the analysis of the front door trim for side impact test, so what need to be followed is a certain flow which will make this project a success. This involves two main things as follows:-

1. Product Knowledge
2. Tools Knowledge

The product knowledge refers to how we know about the product on which we are working. So this needs to be fulfilled by doing literature survey, reading books, going for industrial visit etc. The product knowledge refers to the detailed study of an individual component with how it is manufactured, how it is being fitted, how it gets assembled, etc. The tool knowledge refers to the how we can use different tools to get incorporated with implementation of it. As our project deals with the analysis part, so we have to use different software to get the results. The different tools involving in our project are as follows.

Preprocessor: Hypermesh/ANSA/LSPP
Solver: LS – Dyna
Post Processor: LSPP/Hyper view

2.2 FINITE ELEMENT ANALYSIS

In engineering analysis and design, the finite element approach is used to solve physical problems. The transformation of a physical situation into a mathematical model necessitates a set of assumptions that, when combined, result in differential equations that govern the mathematical model. The physical issue usually entails a real structure or structural component that has been subjected to certain loads. This mathematical model is solved via idealisation element analysis. Because the finite element solution technique is a numerical procedure, the accuracy of the solution must be evaluated. If the accuracy standards are not met, the numerical (i.e., finite element) solution must be repeated with revised solution parameters (such as finer meshes) until the accuracy requirements are met.

HYPERMESH:

Hypermesh is CAE software which comes in after a model has been developed in any CAD software. Hypermesh is preprocessing software where you divide the model into number of elements and nodes for a solver to apply the mathematical functions on it. Hypermesh has advanced model assembly tools for complex sub-system generation and assembly. Modeling of laminate composites is supported by advanced creation, editing and visualization tools. Design changes can be done easily via mesh morphing and geometry dimensioning.

L-S DYNA:

The Livermore Software Technology Corporation developed LS-DYNA, an advanced general-purpose Multi-physics simulation software suite. LS-DYNA is a transient dynamic analysis of extremely nonlinear problems. It is an explicit FE software. The central difference approach is used in LS-DYNA to solve the transient dynamic equilibrium equation. LS-DYNA also uses Newton-Raphson method to solve the non-linear problems, including the contact and impact problems.

HYPERMESH steps for 2D and 3D PART:

1	Import .stp file
2	Delete solids
3	Mid surface extraction
4	2D meshing
5	Mesh refining
6	Quality check

Table no. 2.2.1: Flow Chart – 2D Meshing

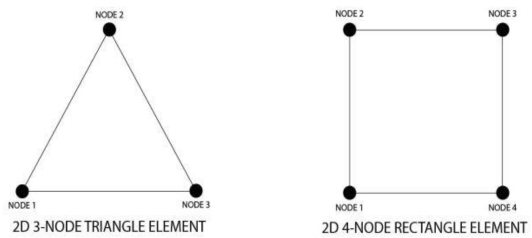
2.3 MESHING:

The geometry has been imported to Hypermesh, by converting .CAT Product into .stp file or .igs file. Solid manual mesh is done in Hypermesh student version software in which 2D mesh and 3D tetra mesh is done. An element size is taken of 5 mm. The meshing carried out is fine. Quality check of the mesh was carried out to achive the maximum accuracy of the results. Meshing is done to convert infinite number of points in continuum into finite number of elements to make calculations easier. Parts have been meshed in Hypermesh.

There are two types of meshing:

2D Meshing:

Two of the dimensions are very large in comparison to third one. Hence for appropriate representation of geometry via 2D Mesh it is necessary to extract mid surface and generate nodes and element on mid surface.



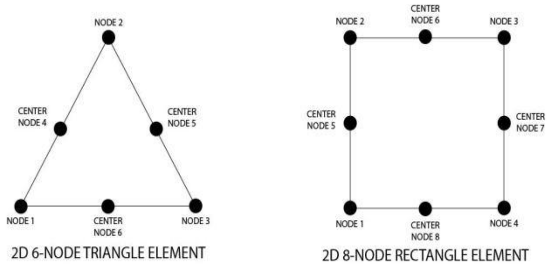


Fig no.2.3.1: 2D Elements

3D Meshing:

The 3D elements should be used when all dimensions are comparable. Hypermesh has several functions that require the definition of volume such as creating tetrahedral meshes. This can be done either with surfaces that encloses the volume of or with solved geometrical entities. In 3D meshing we use tetra element. This element also called as tetrahedron element this element consist of four triangular faces, six straight edges and four vertex corner.

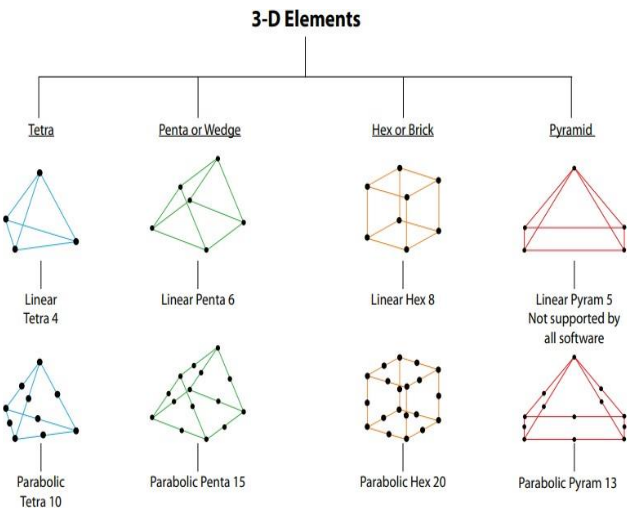


Fig no.2.3.2: 3D Elements

2.4 Meshed Part:

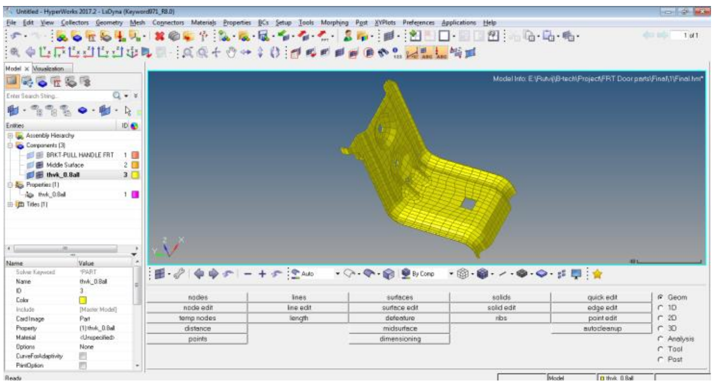


Fig no.2.4.1: Bracket Pull Handle

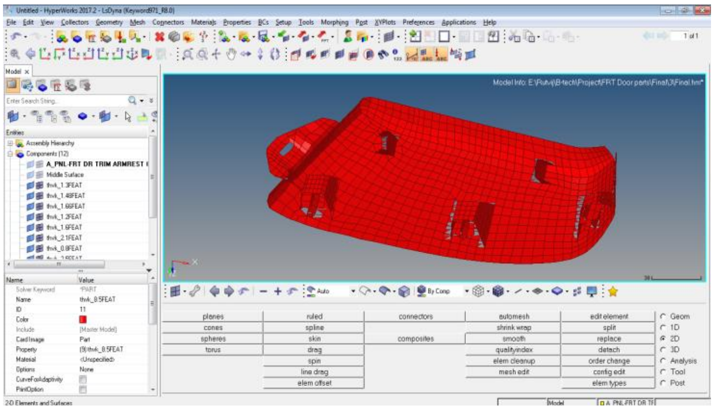


Fig no.2.4.2: Door Arm Rest

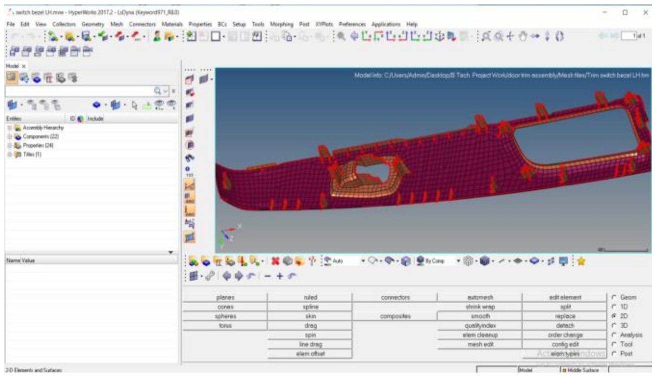


Fig no.2.4.3: Left Hand Switch Bezel

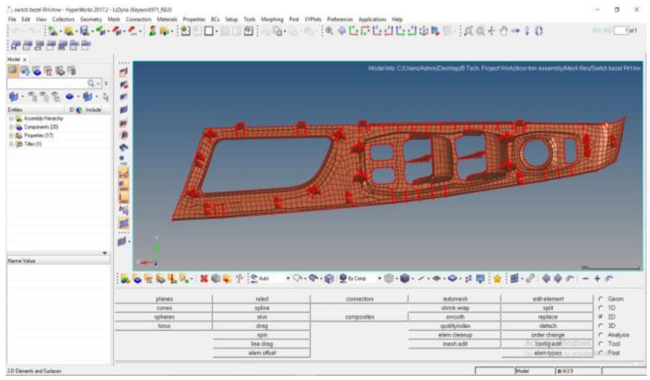


Fig no. 2.4.4: Right Hand Switch Bezel

2.5 Meshing Quality:

Expected defect in meshing should be 0. Defect obtained in meshing 0, Shown by red block in figure 2.5.1

1) Bracket Pull Handle With 0% Defect

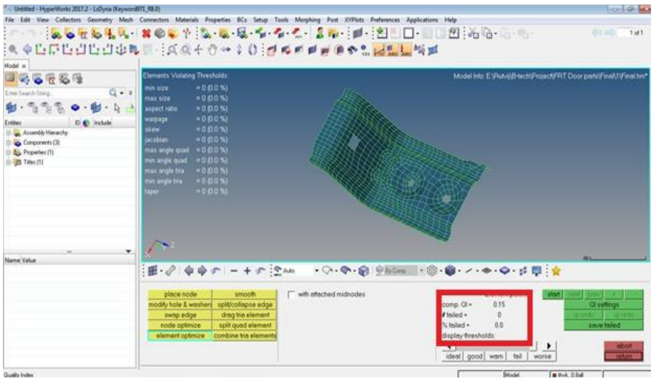


Fig. no. 2.5.1: Bracket Pull Handle With 0% Defect

2) Door Arm Rest With 0% Defect

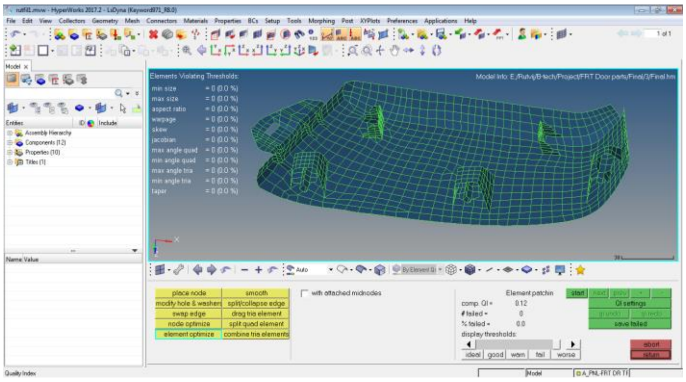


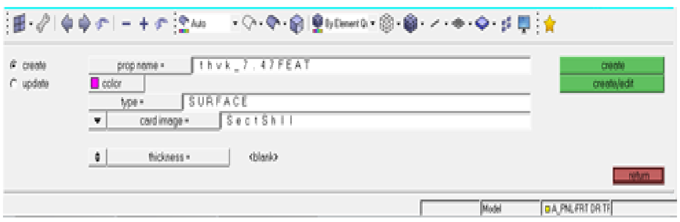
Fig. no. 2.5.2: Door Arm Rest With 0% Defect

2.6 Thickness Assigning to Meshed Parts:

Mathematically element thickness is assigned half in +Z axis (element top) and half in -Z axis (element bottom).

Steps for Assigning Thickness :

1. Create property (prop name, type, card image, thickness):



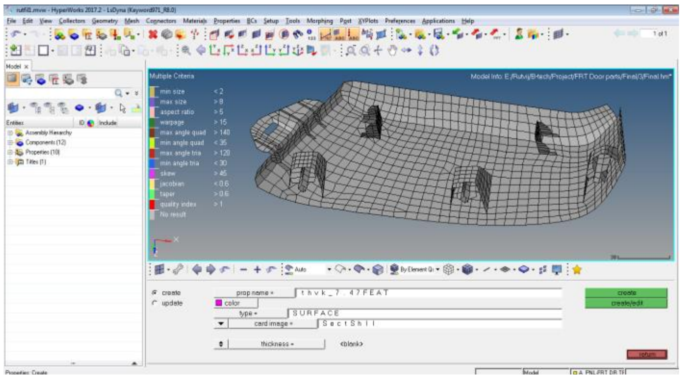


Fig. no. 2.6.1: Create Property

2. Create component (comp name, card image, property):

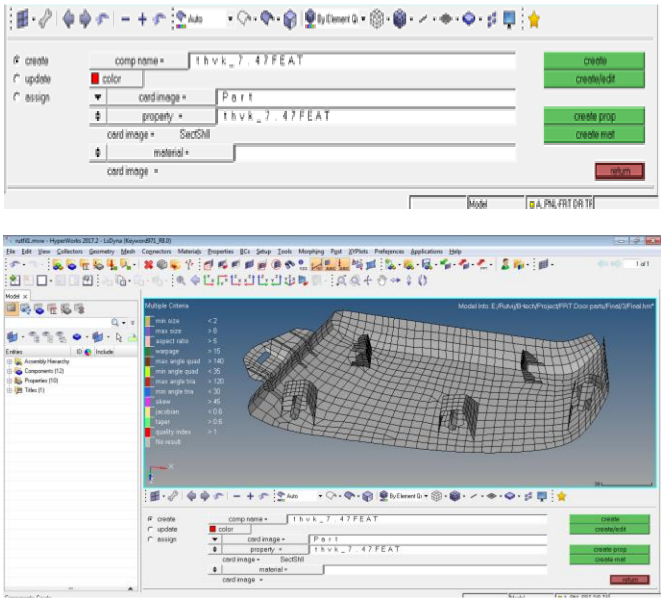
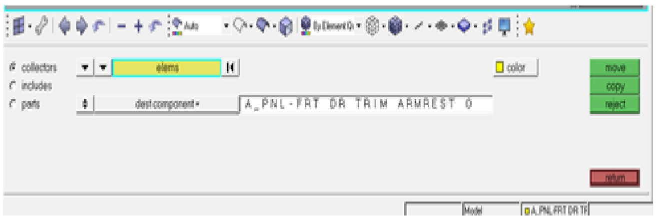


Fig. no. 2.6.2: Create Component

3. Organize Entities:

1) Select elements to which thickness to be assigned.
2) Select the Component which we have created in previous step.
3) Move the selected portion so that thickness is assigned.



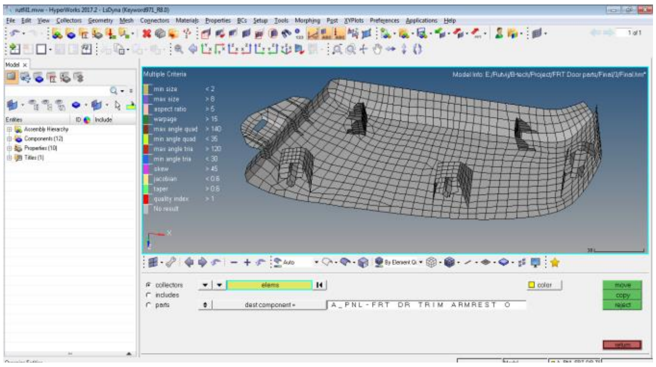


Fig. no. 2.6.3: Organize Entities

4. Finished Part.

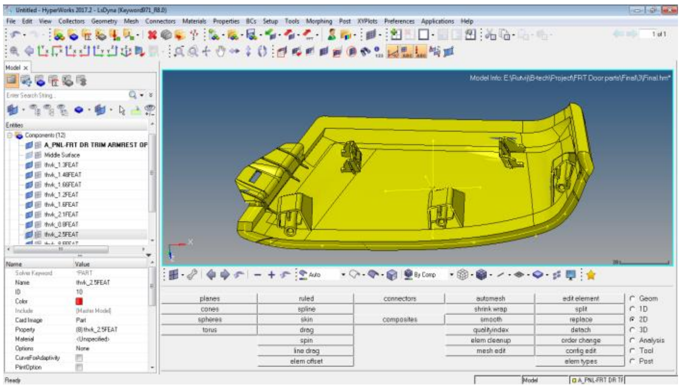


Fig. no. 2.6.4: Door arm rest with thickness

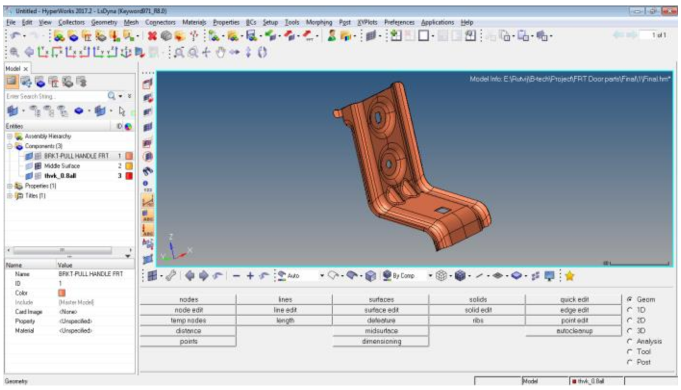


Fig. no. 2.6.5: Bracket Pull Handle with Thickness

2.7 Assembly of Door Trim:

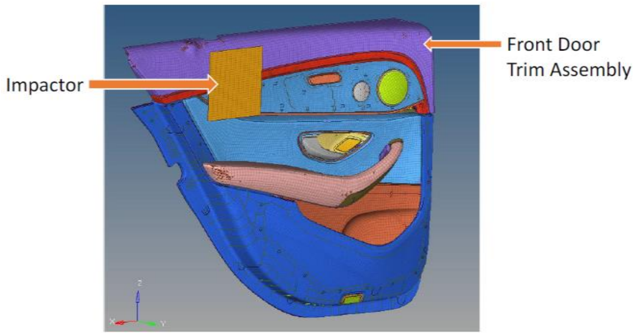


Fig. no. 2.7.1: Assembly of Front Door Trim

3. RESULTS OF EXPLICIT ANALYSIS

Displacement Plot:

Displacement plots are simpler to understand. Displacement plot shows the deformation along the part of the geometry as well as it shows the undeformed area for the part of the geometry. Displacement plots are used to review the mode shapes from a modal analysis. Displacement plot shows the color contour which symbolises the particular range of displacement. The color countour gives maximum and minimum stressed part of the geometry. As in the above shown figure, the maximum displacement is 10.75mm and minimum displacement is 0 mm for undeformed area. In displacement plot the color contour is also shown for an impactor.

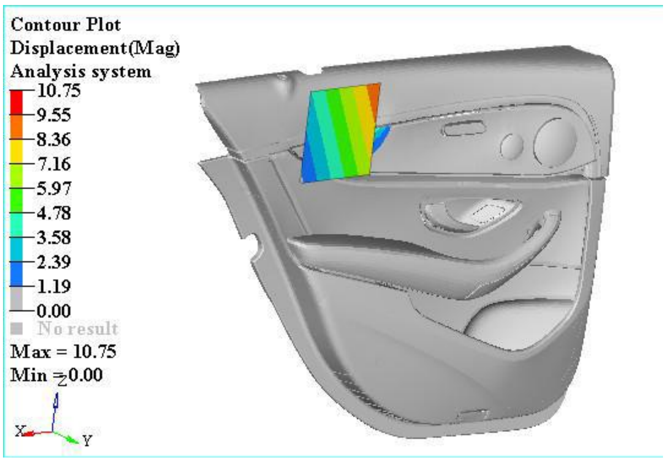


Fig. no.3.1: Displacement Plot

Stress Plot:

Stress plot is similar to displacement plot and simple for understanding. Stress plot shows the part of the geometry where stress occurs due to application of the load. Stress plot shows the color contour which symbolizes the particular range of stress for each color. The color contour gives maximum and minimum stressed part of the geometry. The red color shows about the maximum stressed region which has value of 56.22 MPa. The grey color indicate the minimum stressed region or unstressed region i.e. 0 MPa.

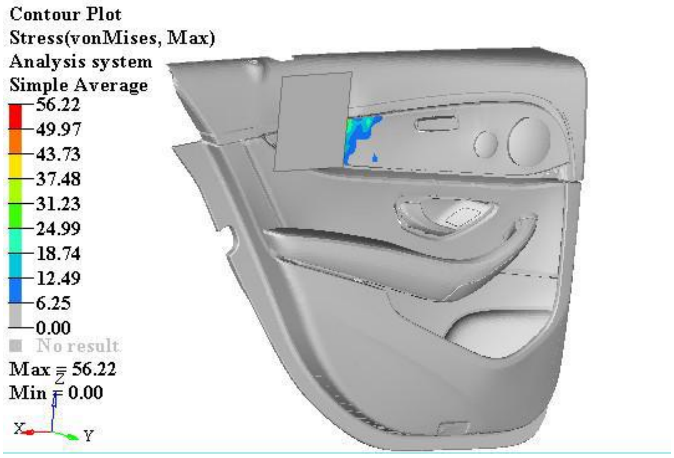


Fig. no.3.2: Stress Plot

Impactor Displacement:

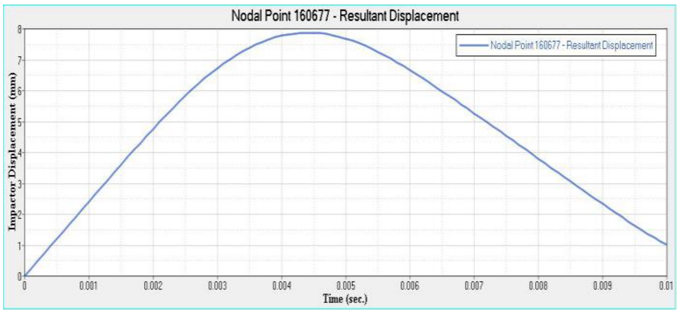


Fig. no.3.3: Impactor Displacement

The fig no. 3.3 shows about how impactor moves along door trim over period of time. When impactor touches the door trim impactor gets displaced. The above graph is also showing the same thing and curve shows upward trend like bell shape curve.

Impactor Reaction Force:
Velocity is given to crash simulation as per standard test procedure which is FMVSS No. 213 and 214. Velocity of impactor is given about 2.4 m/s. We have to analyze the amount of force exerted on door trim part in order to design door trim accordingly. Reaction force is generated in the door trim which is measured using transducer. Peak amount of force which act on trim parts has an approximate magnitude of 19000N. Maximum amount of force acts at 0.004 seconds after initiation of collision. While increase of force magnitude it takes just 0.003 seconds and force magnitude to decrease from maximum to zero takes about 0.006 seconds. Force dissipation takes twice the time of increase in force.

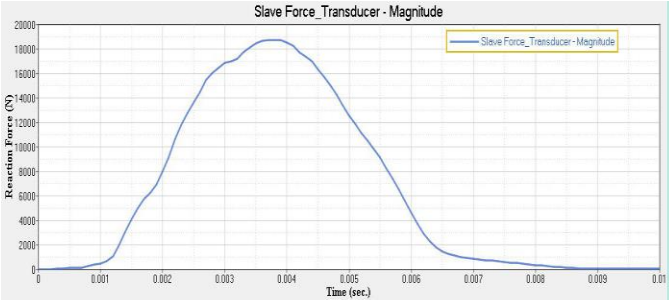


Fig. no. 3.4: Graph illustrating impactor reaction force Vs time

Energy Exertion Behavior:
Amount of energy exerted is an important factor in order to analyze the crash simulations. Energy exerted has some behavior depending upon crash simulation. Mainly three types of energy are governed while side impact analysis. As shown in fig. no. 3.5 kinetic energy of colliding bodies are at maximum in initial condition as body is in motion. After collision velocity of body tends to decrease due to resistance and as a part of reaction impacting body moves in opposite direction. Internal energy is at its minimum value when both bodies are not collided, it tends to increase after collision takes place. Internal energy is maximum at point after maximum collision force between 0.004 to 0.005 seconds. Interface energy has least magnitude among all three energy which has maximum effect at point of maximum reaction force. Maximum internal energy exerted is approximately 45000 MJ

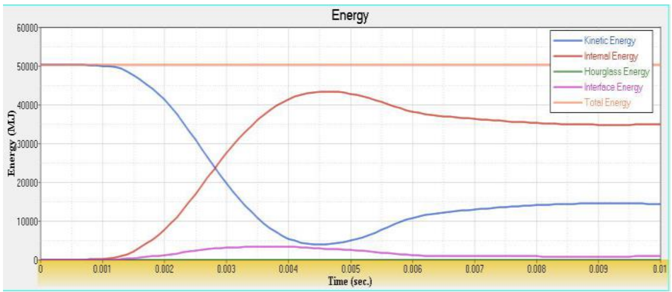


Fig. no. 3.5: Energy Plot

Force Vs Displacement:
Increase in force is found to be in very less time with linear proportional curve against displacement. Increase in force magnitude causes increase in deformation of door trim. Displacement of body is found to be maximum at point when force is at its maximum value.

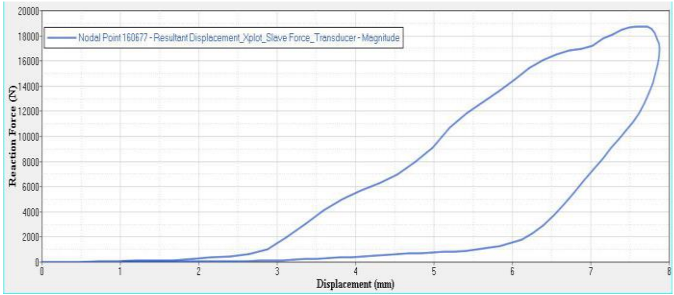


Fig. no. 3.6: Force vs Displacement

4. CONCLUSION:

Explicit analysis of side impact door trim is successfully accomplished. Occupant and side door impact was simulated considering condition of FMVSS No.213 and 214. In this simulation various parameters such as displacement, stress, energy and force were monitored. Different plots were obtained using software tool which illustrated behaviour of material under impact conditions. Impactor forces and reaction forces are generated at interface of door and occupant. Maximum displacement is 10.75mm which is slightly greater than desirable value. Impactor reaction force attains peak value slightly later than impactor peak value. Energy curves are following desired behaviour which can be used to analyse behaviour of system simulation. Peak value of force reaches to approximately 19000N. It can be concluded that energy curve depicts the validated simulation parameters and results. Material properties are not applied on basis of different grades which has proved to be limitation for project.

5. FUTURE SCOPE:

Materials and its properties used in simulation are based on standard and general values. This tends to higher displacement results as properties don‟t match industrial developed materials. Research and development of door trim materials are continuously conducted in order to reduce occupant injury and increase safety. Foam properties are not available with standard references which are used in modern automotive doors. In this project, material research tends to lag behind in entire research work. Future scope for project is to review more materials which are possible.

6. REFERENCES:

[1] Rajesh P. Diwate, Prashant Deshmukh, Developmet of Passanger Vehicle Door Trim for Occupant Saftey by using CAE, International Journal of Scientific & Technology Research Volume, ISSN 2277-8616, Volume 4, Issue 01, January 2015

[2] Reichert R. Kan, Arnold Keifer, Mueller, IIHS (Insurance Institute for Highway Safety) side impact parametric study using LS-DYNA, 15th International LS-DYNA Users Conference, June 2018 [3] Dong Liping, Zhu Xichan, Ma Zhixiong, Study on the door trim intrusion velocity based on occupant injuries of side impact, Biotechnology an Indian Journal, ISSN 0974-7435, Volume 10, Issue 24, 2014

[4] Sandeep Bundele, Rupa Bindu, Automotive door design and structural optimization of front door for commercial vehicle with ULSAB concept for cost and weight reduction, Global Journal of Researches in Engineering Automotive Engineering, ISSN 2S249-4596, Volume 12, Issue 2, 2012

[5] Tawanda Mushiri, Charles Mbohwa, Design of a vehicle door structure based on finite element method, 6th Int Conference on Mechanical, Production and Automobile Engineering, ICMPAE 2014, Nov. 2014

[6] Ali Ghadianlou, Shahrir Bin Abdullah, Crashworthiness design of vehicle side door beams under low speed pole side impact, Thin Walled Structure, Elsevier 2013, Volume 67, March 2013

[7] FMVSS 214, NHTSA 29134